Unsteady RANS Calculations of Perturbed Natural Circulating Flow Within an Experimental Test Loop

By **Deacon Marshall**, **Sophie L Brown**, **Ryan Tunstall**, **Alex Skillen**

SUMMARY

- The implications of transient events can have significant influence on the cooling capabilities of a natural circulation loop, where perturbations such as cold fluid injection can hinder natural circulation flow.
- Comparisons have been made between unsteady Reynolds-averaged Navier-Stokes (RANS) computations to high-fidelity Large Eddy Simulation (LES) results for a simple natural circulation loop.
- Two transient types were considered; low point cold fluid injection for 2000 seconds, and zero power scenarios where the heat input was removed once a steady state was established.
- The results from RANS and LES were shown to be comparable in their predictions of bulk mass flow rate and temperature trends, with both transient types demonstrating a reduction in prevailing mass flow rate and heat transfer capability.

1. INTRODUCTION

The primary function of a natural circulation (NC) loop is to transport heat from a source to a sink, the heat transfer capability of which is dependent on the mass flow rate that it can generate through buoyant effects. Flow within an NC-driven loop is of particular interest in the study of hydraulic phenomena as it exhibits different behaviour to pumped flow. Experimental test loops typically consist of a basic rectangular pipe loop, with a heater located at a lower vertical displacement than the cooler. The flow is driven by the resultant temperature differences, creating a density differential, subsequently producing a buoyant force. For a fixed temperature difference between the heater and cooler, the buoyancy forces will eventually become balanced by hydraulic losses through the pipe, and a steady state will be achieved.

Natural circulation can be used as a passive safety measure in pressurised water reactors (PWRs). During postulated fault scenarios, such as abnormal shutdown, passive NC could be relied upon in the absence of forced convection. It may also be necessary to inject cold fluid during fault conditions such as loss of coolant accidents. During these scenarios it is hypothesised that NC may become unstable or stall; these conditions would result in less effective thermal energy transfer. Hence, reliable prediction of the flow rate is essential for the design and performance evaluation of systems that rely on NC.

Numerically assessing NC under such fault conditions is challenging. Buoyant forces can enhance or supress local turbulence levels, whilst complex three-dimensional (3D) flow behaviour such as thermal stratification and recirculating flow is typical. One-dimensional systems codes are widely used in industry to predict system flow behaviour, however, there are limitations in their ability to accurately predict scenarios where complex 3D flow patterns are prominent. This motivates a modelling approach that is more capable of capturing 3D phenomena associated with perturbed NC.

In an effort to improve on one-dimensional (1D) systems code capabilities, use of 3D computational fluid dynamics (CFD) has been investigated. These modelling methods solve the Navier-Stokes equations over a domain discretised by a mesh comprised of small cells. Although capturing complex 3D flow behaviour and turbulent structures is desirable for this application, high-fidelity CFD approaches which resolve turbulent fluctuations, such as Large Eddy Simulations (LES), are computationally expensive. Alternative CFD modelling methods are often favoured in industry due to their lower computational cost, primarily Reynolds-averaged Navier-Stokes (RANS) simulations where the effects of turbulence are modelled rather than resolved. There are a range of modelling assumptions and closures in RANS simulations which require application specific validation.

This study aims to provide an initial assessment of the capability of RANS CFD in predicting the effects of perturbations to NC flow in a simple loop, by comparing its predictions to those from higher-fidelity wall resolved LES. The LES data presented in this study was provided by the University of Manchester, obtained via the methods outlined in [1]. The cases presented focus on a relatively simple NC loop to enable high-fidelity LES, nevertheless, it is expected to exhibit phenomena of interest akin to real applications.

2. METHODOLOGY

The geometry, boundary conditions and fluid properties used for the unsteady RANS model are consistent with those in LES. To demonstrate sufficient confidence in the unsteady RANS CFD model for the intended purpose, a holistic CFD model evaluation approach was used. This approach considers four key steps.

2.1. Physical Description

A diagram illustrating the geometry of the simple NC loop is shown in Figure 1, dimensions of the test loop are provided in [1]. The geometry comprises pipe sections forming a closed loop, with an inclination along the upper pipework and two U-bends at its base, the lower right-hand of which is referred to as the loop seal. The loop contains a heated section in the lower left pipework, with a cooler located at the upper right of the loop. A T-junction is located on the horizontal pipework of the loop seal, which facilitates the injection of cold fluid. An overflow connection is provided by a T-junction above the heated region, which prevents injected fluid pressurising the loop.

The heater was represented using a uniformly distributed volumetric heat source, as the speculative design relies on heated tube bundles protruding into the bulk flow. Sensitivity studies have been performed where simulations were run with a volumetric heat source and explicitly resolved heaters, demonstrating this approximation to be reasonable. The cooling is intended to be provided via a “cooler jacket”; therefore, the cooler region was modelled with a fixed wall temperature boundary condition. All other pipe walls were treated as adiabatic, with a no-slip boundary condition applied to match the LES modelling in [1]. This included the omittance of conjugate heat transfer within the pipework as the facility is intended to be well-lagged to minimise heat losses to the environment; hand calculations were performed to support the assumption that heat losses will be negligible. Shock losses derived using correlations appropriate for the test loop geometry were imposed at the heater, cooler, and central horizontal pipework between the U-bends to replicate the effects of discrete losses at these locations in a practical NC loop. The shock loss at the heater was distributed across its length, whilst the additional losses at the other locations were imposed over short sections of upstream pipework.

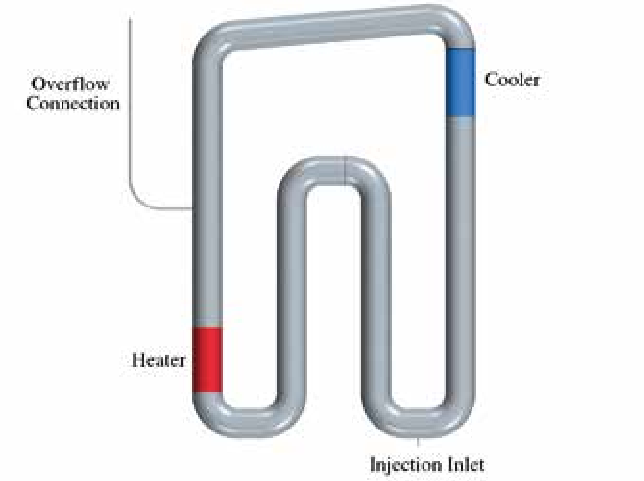


FIGURE 1: Image illustrating the geometry of the NC loop.

2.2. Model Selection

Results were obtained using unsteady RANS simulations, hence turbulent fluctuations are modelled rather than resolved. This means that the solution for each time step effectively corresponds to an ensemble average over many realisations of a particular transient. The Standard K-Epsilon 2-Layer (SKE2L) turbulence model was used as this was found to be the most suitable model available, based on a review of findings from an internal suite of relevant single-effect validation cases. Since NC relies heavily on the thermal driving head, body forces due to gravity have been included to simulate buoyant effects.

The RANS model was discretised using a polyhedral mesher with approximately 4 M cells, whereas the LES model used a blockstructured approach with approximately 150 M cells. Both models used near-wall prism layers to enable fluid boundary layers to be resolved. A circular cross-section of the RANS mesh is illustrated by Figure 2, with a cross-section of the injection inlet T-junction shown in Figure 3. Figure 4 and Figure 5 show the same cross-sections of the LES model.

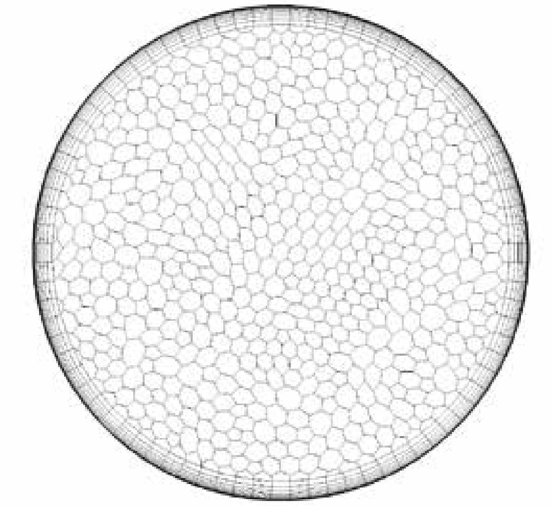


FIGURE 2: Circular cross-section of the RANS mesh in the stream direction.

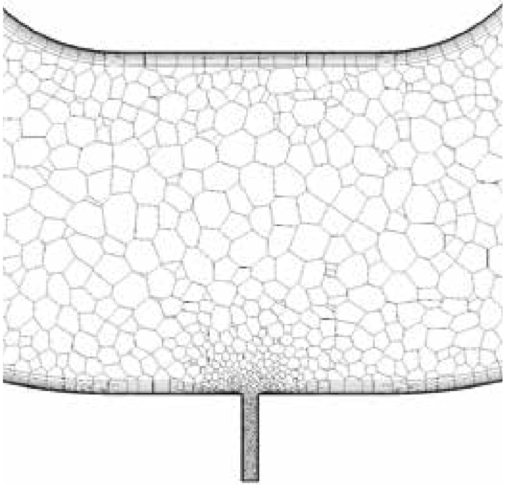


FIGURE 3: Planar cross-section of the RANS mesh at the injection inlet T-junction.

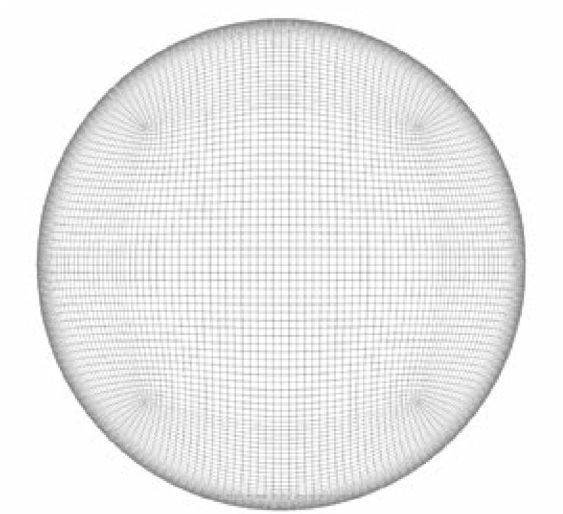


FIGURE 4: Circular cross-section of the LES mesh in the stream direction.

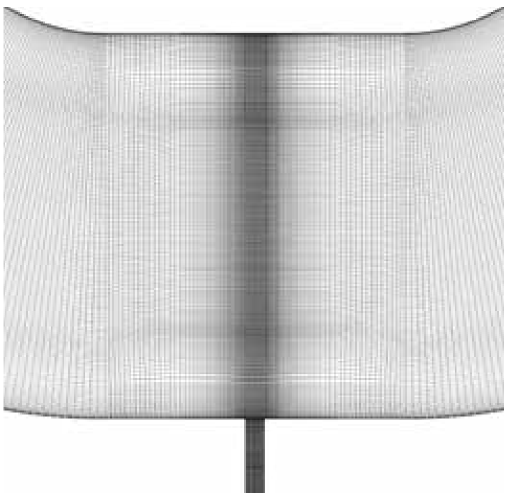


FIGURE 5: Planar cross-section of the LES mesh at the injection inlet T-junction.

Overall, the CFD model is configured in such a way that it has the theoretical capabilities to predict the inherent physics to sufficient resolution and capture the NC flow driven by buoyancy effects within the loop.

2.3. Solution Procedure

The CFD analysis was performed using STAR-CCM+ 17.02.007_R8 (Double Precision). To ensure that physically credible initial conditions were established prior to initiating a transient, a steady state RANS solution was used. Results from this solution were then used to initialise a 100 s precursor unsteady RANS simulation, to achieve statistically steady state NC. During each transient, predicted loop mass flow rates, velocity magnitudes, and temperatures at various locations were monitored to assess numerical convergence within each time-step. A sufficient number of maximum inner iterations with asymptotic stopping criteria were employed to assure this convergence. Adaptive time-stepping was used to ensure sufficient temporal resolution; this was driven by suitable target, mean, and maximum convective Courant-Friedrichs-Lewy numbers. Predicted flow fields and quantities of interest were reviewed to assess their physical credibility. These steps taken ensured that the solutions had converged and that the results were physically credible.

2.4. Sensitivity Studies and Validation

Mesh and time-step sensitivity studies were carried out to ensure that results were insensitive to both; these studies demonstrated that the baseline model settings were appropriate, on the basis of them resulting in negligible discretisation errors. The modelling approaches employed within this work have been validated against relevant experimental test data within literature for relevant phenomena, for example, cases relating to NC in [2] and thermal stratification in [3]. The sensitivity studies and validation cases provide confidence in the results and demonstrate that there was no evidence to undermine the credibility of the modelling approach.

3. RESULTS

Comparisons have been made to the LES results for cases concerning cold fluid injection and zero-power transients. For each case, monitors of mass flow rate and the bulk temperature difference between the heater and cooler (ΔT) have been compared.

3.1 Injection Transient

Cold fluid injection was initiated at a rate of 0.1 kg/s for 2000 s and the prevailing loop mass flow rate was monitored to assess whether this causes the loop NC to stall. The flow was expected to result in a mass flow rate and temperature reduction during the injection period, and subsequently exhibit recovery of NC once the injection stops.

Figure 6 illustrates that both models predict a reduction in bulk flow rate following injection at t = 100 s, tending towards recovery once the injection ceased at t = 2100 s. It is apparent that unsteady RANS does not capture the smaller oscillations in mass flow rate at the same magnitude that LES does, however, the general trend in flow behaviour is closely followed. This is a result of the LES resolving instantaneous turbulent temperature fluctuations, whilst unsteady RANS predicts a Reynolds-Averaged

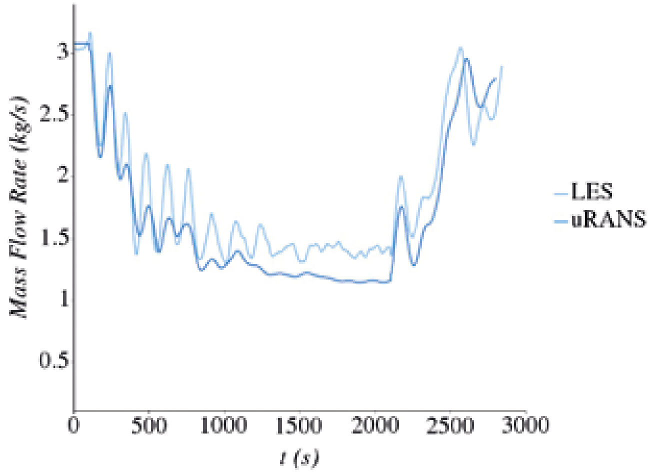


FIGURE 6: Temporal evolution of mass flow rate, where cold fluid was injected at a rate of 0.1 kg/s from t = 100 s to t = 2100 s.

flow field. Figure 7 shows the ΔT behaviour in both approaches to follow a similar trend, with LES predicting stronger oscillatory behaviour which coincides with fluctuations in mass flow.

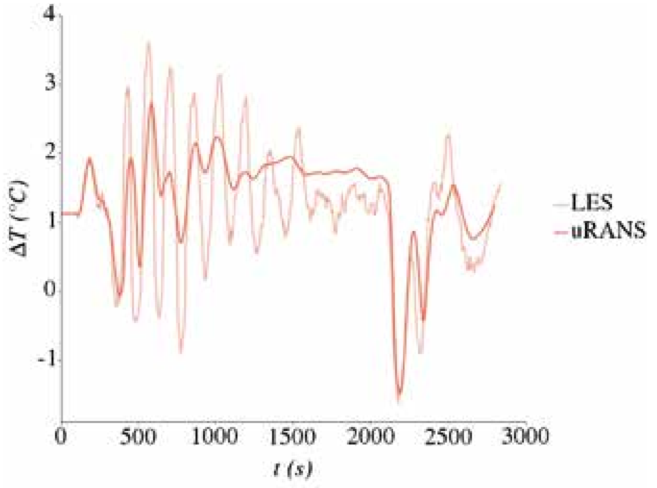


FIGURE 7: Temporal evolution of ΔT, where cold fluid was injected at a rate of 0.1 kg/s from t = 100 s to t = 2100 s.

Figure 10 shows temperature contours in the loop seal region on the x-y plane, at various times during cold fluid injection. These contours demonstrate the evolution of cold-trap formation in the Ubend, gradually resulting in thermal stratification of the flow in the lower horizontal pipe section. The stagnant fluid in the U-bend illustrated in Figure 11 results in higher flow resistance and effectively acts as a reduction in hydraulic diameter, contributing to an overall decrease in mass flow rate as shown in Figure 6. Between t = 2200 s and 2500 we observe a diminished stratification height following the termination of cold fluid injection, due to the stratified fluid being cleared as a result of NC recovery.

3.2. Zero Power Transient

The heater power was ramped down to 0 kW between t = 100 s and 101 s after establishing statistically steady state NC. The mass flow rate and temperature were expected to decrease due to a significant reduction in the thermal driving head. Figure 8 shows both approaches predict a sudden reduction in mass flow rate following the removal of heat input, with subsequent decaying oscillations. As seen with the injection transient, LES predicts greater oscillations in mass flow rate than unsteady RANS. Figure 9 illustrates that LES also predicts oscillations in ΔT with a greater

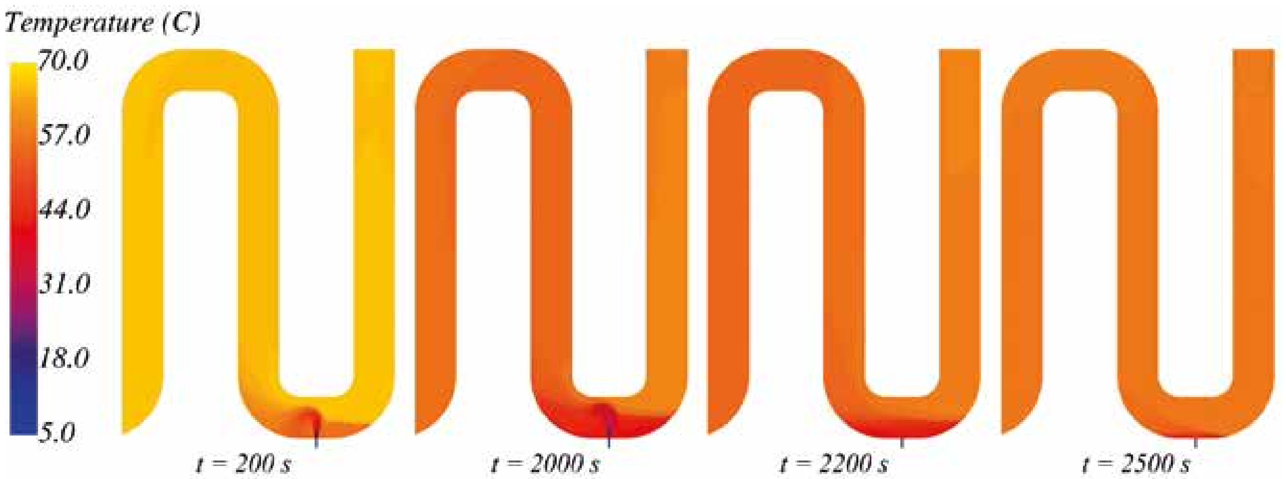


FIGURE 10: Intermittent temperature contour plots of the loop seal, where cold fluid was injected at a rate of 0.1 kg/s from t = 100 s to t = 2100 s.

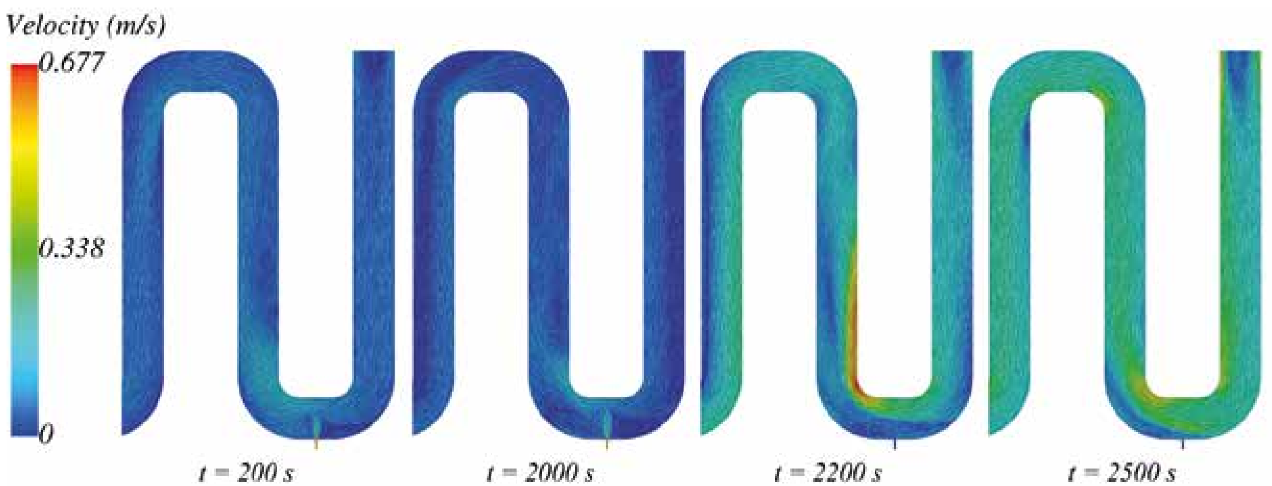


FIGURE 11: Intermittent velocity contour plots of the loop seal, where cold fluid was injected at a rate of 0.1 kg/s from t = 100 s to t = 2100 s.

magnitude, consistent with those observed for mass flow rate.

Discrepancies in the magnitude of oscillations during intermittent NC recovery is speculated to be linked to cold flow over the top of the U-bend. This may be a result of RANS employing a two-layer all-y+ wall treatment as opposed to the wall-resolved LES, leading to the flow remaining attached to the wall and significantly altering the local buoyant force. The greater reduction in flow rate observed with LES allows for more heat transfer in the fluid at the cooler, resulting in the local buoyant force overcoming the stalled/recirculating fluid before the Ubend. Buoyant effects cause cold fluid to fall through the downcomer, leading to greater magnitudes of intermittent mass flow rate recovery.

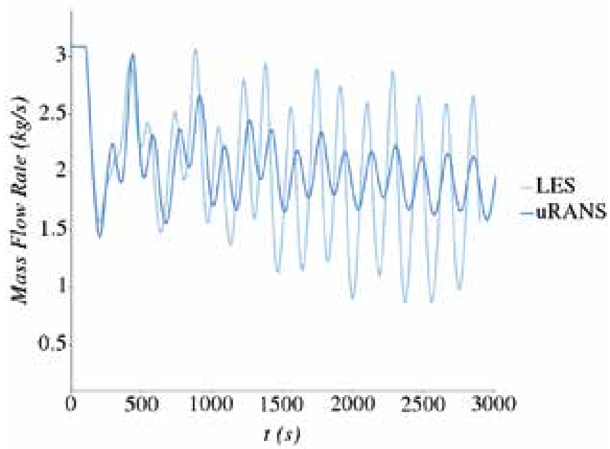


FIGURE 8: Temporal evolution of mass flow rate, where the heater was switched off at t = 100 s.

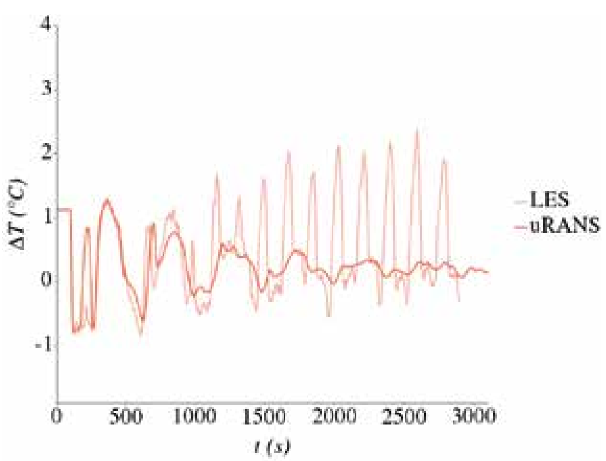


FIGURE 9: Temporal evolution of ΔT, where the heater was switched off at t = 100 s.

4. CONCLUSIONS

Unsteady RANS simulations of perturbed flow within an NC driven pipe loop have been performed. For the cold fluid injection transient, both LES and unsteady RANS approaches predict a reduction in loop mass flow rate and a decrease in ΔT between the heater and cooler. Once injection ceased, both approaches predict recovery in NC flow, tending towards the initial steady conditions. Results from the zero power transient simulations also show that both CFD modelling approaches predict a net decrease in mass flow rate and ΔT. The consequences of RANS failing to predict oscillations in mass flow and ΔT of similar magnitude to LES are insignificant for this study, as the results follow the same general trend as the loop cools.

This work has demonstrated good agreement between general trends predicted by unsteady RANS and LES for the perturbed NC scenarios considered, however, LES was shown to predict greater oscillations in mass flow rate and ΔT. This is expected due to the ability of LES to resolve turbulent fluctuations within the flow field. These comparisons suggest that RANS is a promising 3D modelling approach for predicting general trends when NC driven flow is perturbed, offering a route to mitigate the higher computational cost associated with LES.

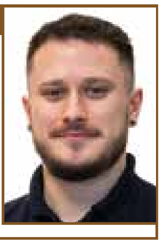
REFERENCES

[1] A. Skillen, R. Tunstall, S. Parry, S. Hind, S. Treasure, Natural Convection Within a Loop With Cold Flow Injection, UK Heat Transfer Conference, 4-6 April 2022, Manchester, UK.

[2] Betts, P., & Bokhari, I. "Experiments on turbulent natural convection in an enclosed tall cavity", University of Manchester Institute of Science and Technology, 2000.

[3] P. L. Viollet, Observation and numerical modelling of density currents resulting from thermal transients in a non-rectilinear pipe, Journal of Hydraulic Research, Vol 25, No 2, p235-261, 1987.

DEACON MARSHALL



Deacon holds the position of Thermofluid Engineer at Rolls-Royce plc, his focus being in developing computationally feasible analysis methods to substantiate fluid system safety. He has a BEng (Hons) in Aerospace Engineering from Sheffield Hallam University with a thesis on CFD analysis

DR SOPHIE L BROWN

Sophie Brown completed her PhD in Applied Aerodynamics at Loughborough University in 2019. Following this she worked as an Aerodynamics Engineer in the Automotive Industry before joining Rolls-Royce as a Thermofluid Engineer in 2020.

DR RYAN TUNSTALL

Dr Ryan Tunstall has a PhD in CFD and turbulence modelling for nuclear power and propulsion systems from the University of Manchester and is a Chartered Engineer with the IMechE.,He has spent over eight years at Rolls-Royce plc, where he is a Technical Specialist in Nuclear Thermal Hydraulics.

DR ALEX SKILLEN

Dr. Alex Skillen is a Lecturer in Engineering Simulation and Data Science within the Department of Mechanical and Aerospace Engineering at the University of Manchester. Alex's research interests lie at the intersection of Computational Fluid Dynamics (CFD) and machine learning.